

NUMERICAL SOLUTION OF THREE DIMENSIONAL CONJUGATE HEAT TRANSFER IN A MICROCHANNEL HEAT SINK AND CFD ANALYSIS

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology
in
Mechanical Engineering**

BY

AMRITRAJ BHANJA



**DEPARTMENT OF MECHANICAL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA-769008
2009**

NUMERICAL SOLUTION OF THREE DIMENSIONAL CONJUGATE HEAT TRANSFER IN A MICROCHANNEL HEAT SINK AND CFD ANALYSIS

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology
in
Mechanical Engineering**

BY

AMRITRAJ BHANJA

Under the guidance of

Dr. ASHOK KUMAR SATAPATHY



**DEPARTMENT OF MECHANICAL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA-769008
2009**



**NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA**

CERTIFICATE

This is to certify that the thesis entitled **“NUMERICAL SOLUTION OF THREE DIMENSIONAL CONJUGATE HEAT TRANSFER IN A MICROCHANNEL HEAT SINK AND CFD ANALYSIS”** submitted by Sri Amritraj Bhanja in partial fulfillment of the requirements for the award of Bachelor of technology Degree in Mechanical Engineering at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University / Institute for the award of any Degree or Diploma.

DATE:

DR. ASHOK KUMAR SATAPATHY
DEPARTMENT OF MECHANICAL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA-769008



National Institute of Technology Rourkela

ACKNOWLEDGEMENT

I would like to express my deep sense of gratitude and respect to my supervisor Prof. Ashok Kumar Satapathy, for his excellent guidance, suggestions and constructive criticism. I consider myself extremely lucky to be able to work under the guidance of such a dynamic personality.

I would like to render heartiest thanks to our M.Tech students(ME) whose ever helping nature and suggestion has helped us to complete this present work

DATE:

AMRITRAJ BHANJA
ROLL NO 10503071, B.TECH
DEPARTMENT OF MECHANICAL ENGINEERING
NATIONAL INSTITUTE OF TECHNOLOGY
ROURKELA-769008

CONTENTS

Contents.....	i
Abstract.....	ii
List of Figures.....	iii
1. Introduction	
1.1 Micro-channel Heat Sink.....	2
1.2 About the Project.....	4
1.3 GAMBIT-FLUENT	5
2. Theory	
2.1 Heat Transfer by Conduction	8
2.2 Heat Transfer by Radiation.....	8
2.3 Heat Transfer by Convection.....	8
2.3.1 Reynolds Number.....	9
2.3.2 Nusselt Number.....	9
2.3.3 Prandtl Number.....	9
2.3.4 Grashof Number.....	10
2.3.5 Rayleigh Number.....	10
2.4 Computational Fluid Dynamics & FLUENT	10
3. Design of SINGLE Micro-channel in GAMBIT.....	14
4. Fluent analysis of Micro-channel.....	18
5. Results and Discussions.....	24
6. Conclusion.....	36
7. Application of CFD.....	37
Bibliography.....	38

ABSTRACT

In electronic equipments, thermal management is indispensable for its longevity and hence, it is one of the important topics of current research. The dissipation of heat is necessary for the proper functioning of these instruments. The heat is generated by the resistance encountered by electric current. This has been further hastened by the continued miniaturization of electronic systems which causes increase in the amount of heat generation per unit volume by many folds. Unless proper cooling arrangement is designed, the operating temperature exceeds permissible limit. As a consequence, chances of failure get increased.

Increasing circuit density is driving advanced cooling systems for the next generation microprocessors. Micro-Channel heat exchangers (MHE) in silicon substrates are one method that is receiving considerable attention. These very fine channels in the heat exchanger provide greatly enhanced convective heat transfer rate and have been shown to be able to meet the demands of the cooling challenge for the microprocessors for many generations to come.

This work focused on laminar flow ($Re < 200$) within rectangular micro-channel with hydraulic diameter $86\mu\text{m}$ for single-phase liquid flow. The influence of the thermo-physical properties of the fluid on the flow and heat transfer, are investigated by evaluating thermo-physical properties at a reference bulk temperature. The micro-heat sink model consists of a 10 mm long silicon substrate, with rectangular micro-channels, $57\mu\text{m}$ wide and $180\mu\text{m}$ deep, fabricated along the entire length. Water at 293K is taken as working fluid. The results indicate that thermo-physical properties of the liquid can significantly influence both the flow and heat transfer in the micro-channel. Assumption of hydrodynamic, fully developed laminar flow is valid here on basis of Langhaar's equation. The local heat transfer coefficient and averaged Nusselt number is calculated and plotted for pressure drop of 50kpa, 30kpa and 10kpa. The result is verified for heat flux $50\text{W}/\text{cm}^2$, $90\text{W}/\text{cm}^2$ and $150\text{W}/\text{cm}^2$. A three-dimensional Computational Fluid Dynamics (CFD) model was built using the commercial package, GAMBIT-FLUENT, to investigate the conjugate fluid flow and heat transfer phenomena.

LIST OF FIGURES

Fig 1	Structure of Micro-channel Heat Sink	Page 5
Fig 2	Schematic diagram of single Micro-channel	Page 25
Fig 3	Temperature contour of Outlet at 50 w/cm ² , 50 KPA	Page 25
Fig 4	Temperature contour of Outlet at 90 w/cm ² , 50 KPA	Page 26
Fig 5	Temperature contour of Outlet at 150 w/cm ² , 50 KPA	Page 26
Fig 6	Velocity Contour at 50 KPa	Page 27
Fig 7	Graph of Nu vs Z at 50 Kpa	Page 27
Fig 8	Graph of Nu vs Re at 50 Kpa	Page 28
Fig 9	Temperature contour of Outlet at 50 w/cm ² , 30 KPA	Page 28
Fig 10	Temperature contour of Outlet at 90 w/cm ² , 30 KPA	Page 29
Fig 11	Temperature contour of Outlet at 150 w/cm ² , 30 KPA	Page 29
Fig 12	Velocity Contour at 30 KPa	Page 30
Fig 13	Graph of Nu vs Re at 30 Kpa	Page 30
Fig 14	Graph of Re vs Z	Page 31
Fig 15	Schematic diagram of double Micro-channel	Page 31
Fig 16	Temperature contour of Outlet at 50 w/cm ² , 50 KPA, adiabatic wall conditions	Page 32
Fig 17	Temperature contour of Outlet at 50 w/cm ² , 50 KPA, Isothermal wall conditions	Page 32
Fig 18	Velocity Contour of Outlet	Page 33

CHAPTER 1

INTRODUCTION

- MICROCHANNEL HEAT SINK
- ABOUT THE PROJECT
- GAMBIT-FLUENT

1.INTRODUCTION

1.1. MICROCHANNEL HEAT SINKS:

Advance in micromachining technology in recent years has enabled the design and development of miniaturized systems, which opens a promising field of applications, particularly in the medical science and electronic-/bioengineering. Such systems often contain small scale fluid channels embedded in the surrounding solids with heating sources. Depending on the channel height, it can be described as a **Mini-channel** (at a characteristic dimension of about 1 mm) or a **Micro-channel** (at a characteristic dimension of several microns to several hundred microns). Because of its undeniable advantages of smaller physical dimensions and higher heat transfer efficiency, the study of micro-channel flows has become an attractive research topic with a fast-growing number of publications. Micro-scale heat transfer has received much interest as the size of the devices decreases, such as in electronic equipments, since the amount of heat that needs to be dissipated per unit area increases. The performance of these devices is directly related to the temperature; therefore it is a crucial issue to maintain the electronics at acceptable temperature levels. Micro-channel heat sinks have become known as one of the effective cooling techniques. The design of next generation, giga-scale integrated circuits, requires effective cooling for reliable, long term operations. These devices may even experience catastrophic failure due to generation of very high heat flux (VHHF) transients. The VHHF transients originate from hot spots which may be difficult to localize at all times and will be difficult to handle using conventional cooling techniques. Micro-channel heat-sinks have thus emerged as a promising cooling solution due to high heat transfer coefficient derived from a large surface area to volume ratio.

Micro-channel cooling technology was first put forward by Tuckerman and Pease[1]. They circulated water in micro-channel fabricated in silicon chips and were able to reach a heat flux of 790w/cm^2 without a phase change in a pressure drop of 1.94 bar. Shortly after, Wu and Little [2] obtained experimental results

for fluid flow in micro-channels for gas flow. Their measured friction factors in the laminar regime were higher than expected and they found that the transition Reynolds number ranged from 350 to 900. Peng [3] conducted experiments on micro-channels and they found that the laminar-to-turbulent transition period occurred at Reynolds numbers which were lower than expected from conventional theory. Peng and Peterson found disparities between conventional flow theory and experimental results for micro-channels. They tested micro-channels with hydraulic diameters ranging from 133 μm to 367 μm , and they showed a friction factor dependence on hydraulic diameter and channel aspect ratio [4]

Judy et al. [5] did pressure drop experiments on both round and square micro-channels with hydraulic diameters ranging from 15 to 150 μm . They tested distilled water, methanol and iso-propanol over a Reynolds number range of 8 to 2300. Their results showed no distinguishable deviation from laminar flow theory for each case. Liu and Garimella [6] conducted flow visualization and pressure drop studies on micro-channels with hydraulic diameters ranging from 244 to 974 μm over a Reynolds number range of 230 to 6500. They compared their pressure drop measurements with numerical calculations. Computations were performed for different total pressure drops in the channel to obtain the relation between the overall averaged Nu number and the Re number for this specific geometry of heat sinks. Choi et al. [7] also suggested by their experiments with micro-channels inside diameters ranging from 3 to 81 μm that the Nusselt number did in fact depend on the Reynolds number in laminar micro-channel flow. Experimental measurements for pressure drop and heat transfer coefficient were done by rahman[8]. He used water as a working fluid and tests were conducted on channels of different depths.

Fedorov and Viskanta [9] investigate the conjugate heat transfer in a microchannel heat sink .they found that the average channel wall temperature along the flow direction was nearly uniform except in the region close to the channel inlet, where very large temperature gradients were observed.

1.2. ABOUT THE PROJECT:

In the present study a numerical model with fully developed flow is presented and used to analyze a three dimensional micro-channel heat sink for Re numbers (< 160) and with hydraulic diameter $86\ \mu\text{m}$. In the current investigation, three different cases ($Q_w = 90, 150, \text{ and } 180\ \text{W/cm}^2$, $\Delta P = 50, 15, \text{ and } 10\ \text{kPa}$) were considered. Also, it was analyzed for a heat flux of 50, 90 and $150\ \text{W/cm}^2$. The numerical model is based on three dimensional conjugate heat transfer FLUENT, a commercial package employing continuum model of Navier Stokes equation with SIMPLE algorithm. The numerical solver codes are well-established and thus provide a good start to more complex heat transfer and fluid flow problems. FLUENT provides adaptability to variation of thermo physical properties with respect to temperature effect. The thermo physical properties are chosen at a reference temperature. A series of calculations were carried out to analyze rectangular silicon-based micro-channel heat sink. Computations were performed for different total pressure drops in the channel to obtain the relation between the overall averaged Nu number and the Re number for this specific geometry of heat sinks.

Here the micro-heat sink model consists of a 10 mm long substrate and dimension of rectangular micro-channels have a width of $57\ \mu\text{m}$ and a depth of $180\ \mu\text{m}$ as shown in fig [1]. The heat sink is made from silicon and water is used as the cooling fluid. The electronic component is idealized as a constant heat flux boundary condition at the heat sink bottom wall. Heat transport in the unit cell is a conjugate problem which combines heat conduction in the solid and convective heat transfer to the coolant (water). Here we consider a rectangular channel of dimension ($900\ \mu\text{m} \times 100\ \mu\text{m} \times 10\ \text{mm}$) applied constant heat flux of $90\ \text{W/cm}^2$ from bottom. Water flows through channel at temperature 293K on account of pressure loss of 50kpa.

Here we assumed to have a constant heat flux, q'' ($90\ \text{W/cm}^2$) at the bottom wall. The other wall boundaries of the solid region are assumed to be either perfectly insulated with zero heat flux or at isothermal conditions. The water flow

velocities are taken from different Reynolds numbers, from 96 to 164, with reference to the hydraulic diameter of $86\text{ }\mu\text{m}$.

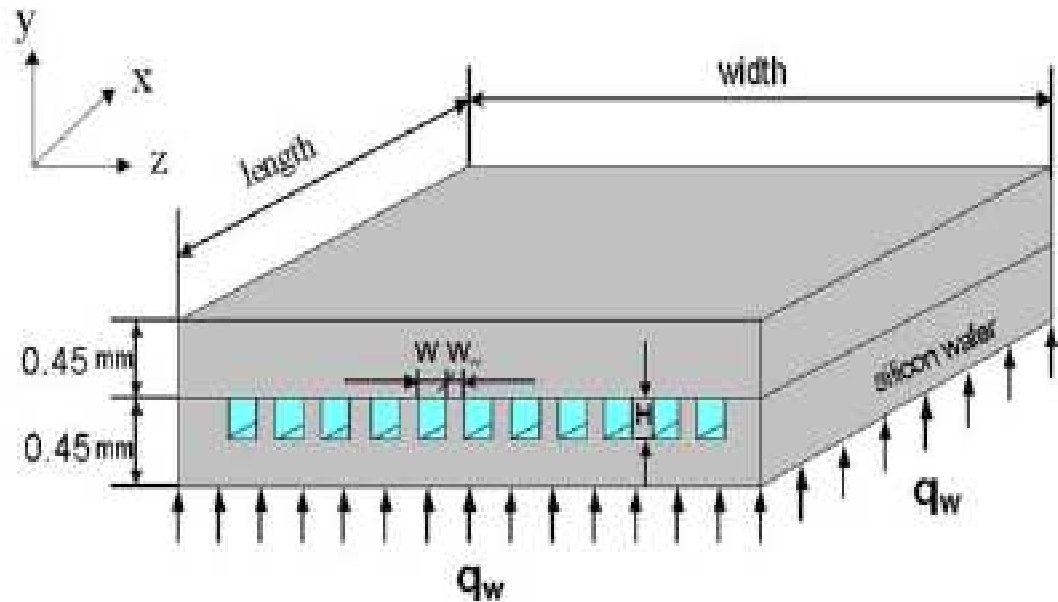


FIG. 1 Structure of Micro-channel Heat Sink

1.3. GAMBIT-FLUENT:

Preprocessing is the first step in building and analyzing a flow model. It includes building the model (or importing from a CAD package), applying the mesh, and entering the data. We used Gambit as the preprocessing tool in our project. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI).

The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications. The advantages of using gambit for the geometry design are:

- Ease of use
- CAD/CAE Integration
- Fast Modeling
- CAD Cleanup
- Intelligent Meshing.

FLUENT is a computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also allows you to refine or coarsen your grid based on the flow solution.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful computer servers. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface.

CHAPTER 2

THEORY

- HEAT TRANSFER
- CFD & FLUENT

2. THEORY

2.1. HEAT TRANSFER:

Heat is defined as energy transferred by virtue of temperature difference or gradient. Being a vector quantity, it flows with a negative temperature gradient. In the subject of heat transfer, it is the rate of heat transfer that becomes the prime focus. The transfer process indicates the tendency of a system to proceed towards equilibrium. There are 3 distinct modes in which heat transfer takes place:

2.1.1. Heat transfer by Conduction:

Conduction is the transfer of heat between 2 bodies or 2 parts of the same body through molecules. This type of heat transfer is governed by Fourier's Law which states that – "Rate of heat transfer is linearly proportional to the temperature gradient". For 1-D heat conduction-

$$q_k = -k \frac{dT}{dx}$$

2.1.2. Heat transfer by Radiation:

Thermal radiation refers to the radiant energy emitted by the bodies by virtue of their own temperature resulting from the thermal excitation of the molecules. It is assumed to propagate in the form of electromagnetic waves and doesn't require any medium to travel. The radiant heat exchange between 2 gray bodies at temperature T₁ and T₂ is given by:

$$Q_{1-2} = \sigma A_1 F_{1-2} (T_1^4 - T_2^4)$$

2.1.3. Heat transfer by Convection:

When heat transfer takes place between a solid surface and a fluid system in motion, the process is known as Convection. When a temperature difference

produces a density difference that results in mass movement, the process is called Free or Natural Convection.

When the mass motion of the fluid is carried by an external device like pump, blower or fan, the process is called Forced Convection. In convective heat transfer, Heat flux is given by:

$$q(x) = h_x(T_w - T_\infty)$$

2.1.3.1. Reynold's Number:

It is the ratio of inertial forces to the viscous forces. It is used to identify different flow regimes such as Laminar or Turbulent.

$$R_e = \frac{\rho V D}{\mu} = \frac{V D}{\nu} = \frac{\text{Inertial Forces}}{\text{Viscous Forces}}$$

2.1.3.2. Nusselt Number:

The nusselt number is a dimensionless number that measures the enhancement of heat transfer from a surface that occurs in a real situation compared to the heat transferred if just conduction occurred.

$$Nu_L = \frac{hL}{k_f}$$

2.1.3.3. Prandtl Number:

It is the ratio of momentum diffusivity (viscosity) and thermal diffusivity.

$$Pr = \frac{\nu}{\alpha}$$

2.1.3.4. Grashof Number:

It is the ratio of Buoyancy force to the viscous force acting on a fluid.

$$Gr = \frac{g\beta (T_s - T_\infty)L^3}{\nu^2}$$

2.1.3.5. Rayleigh Number:

It is the product of Grashof number and Prandtl number. It is a dimensionless number that is associated with buoyancy driven flow i.e. Free or Natural Convection. When the Rayleigh number is below the critical value for that fluid, heat transfer is primarily in the form of conduction; when it exceeds the critical value, heat transfer is primarily in the form of convection.

$$Ra_x = Gr_x Pr = \frac{g\beta}{\nu\alpha}(T_s - T_\infty)x^3$$

2.2. COMPUTATIONAL FLUID DYNAMICS (CFD) & FLUENT:

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations.

2.2.1. Governing equations:

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = \frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{V} = 0$$

Conservation of Momentum equation for an incompressible fluid is:

$$\rho \frac{Du}{Dt} = \rho g_x - \frac{\partial P}{\partial x} + \mu (\Delta^2 u)$$

Energy Equation:

$$\left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = \frac{1}{\alpha} \left(\frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)$$

2.2.2. Discretization Methods:

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are:

2.2.2.1. FINITE VOLUME METHOD:

This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. This integral approach yields a method that is inherently conservative (i.e., quantities such as density remain physically meaningful)

$$\frac{\partial}{\partial t} \iiint Q dV + \iint F d\mathbf{A} = 0$$

Where Q is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier-Stokes equations), V is the cell volume, and A is the cell surface area.

2.2.2.2. FINITE ELEMENT METHOD:

This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. In this method, a weighted residual equation is formed:

$$R_i = \iiint V^e W_i Q dV$$

Where R_i is the equation residual at an element vertex i , Q is the conservation equation expressed on an element basis, W_i is the weight factor and V^e is the volume of the element.

2.2.2.3. FINITE DIFFERENCE METHOD:

This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

Where Q is the vector of conserved variables, and F , G , and H are the fluxes in the x , y , and z directions respectively.

2.2.3. Schemes to Calculate Face Value for variables:

2.2.3.1. Central Difference scheme:

This scheme assumes that the convective property at the interface is the average of the values of its adjacent interfaces.

2.2.3.2. The UPWIND scheme:

According to the upwind scheme, the value of the convective property at the interface is equal to the value at the grid point on the upwind side of the face. It has got 3 sub-schemes i.e. 1st order upwind which is a 1st order accurate, 2nd order upwind which is 2nd order accurate scheme and QUICK (Quadratic Upwind Interpolation for convective kinematics) which is a 3rd order accurate scheme.

2.2.3.3. The EXACT solution:

According to this scheme, in the limit of zero Peclet number, pure diffusion or conduction problem is achieved and ϕ vs x variation is linear. For positive values of P , the value of ϕ is influenced by the upstream value. For large positive values, the value of ϕ seems to be very close to the upstream value.

2.2.3.4. The EXPONENTIAL scheme:

When this scheme is used, it produces exact solution for any value of Peclet number and for any value of grid points. It is not widely used because exponentials are expensive to compute.

2.2.3.5. The HYBRID scheme:

The significance of hybrid scheme can be understood from the fact that (i) it is identical with the Central Difference scheme for the range $-2 \leq Pe \leq 2$ (ii) outside, it reduces to the upwind scheme.

CHAPTER 3

MICROCHANNEL MODELING USING GAMBIT

DESIGN OF SINGLE MICROCHANNEL

USING GAMBIT

STEP 1 Select a solver:

Main Menu>Solver>Fluent 5/6.

STEP 2 Creation of Vertices:

Operation>Geometry>Vertex Command Button>Create Vertex from Co-ordinates.

The following vertices with the required Cartesian co-ordinates were created. All dimensions are in 'um'

<u>Vertices</u>	<u>Cartesian Co-ordinates</u>
A	-50,-450,0
B	50,-450,0
C	50,450,0
D	-50,450,0
P	-28.5,0,0
Q	-28.5,-180,0
R	28.5,-180,0
S	28.5,0,0
O	50,-450,10000

STEP 3 Creation of Line:

Operation>Geometry>Creating Edge>Straight.

Create straight lines by joining the following vertices
AB,BC,CD,DA,PQ,QR,RS,SP,DO.

STEP 4 Creation of Faces:

Operation>Geometry>Face Command Button>Form Face>Create Face from Wireframe.

To create face 'ABCD', select edges in a sequence order (**AB>BC>CD>DA**) and click apply. Similarly, create **PQRS** using the same above commands.

STEP 5 Meshing Edges (PQ, QR, RS, SP):

(a) **Operation>Mesh>Mesh edges.**

Select Edge-PQ,RS

Grading-Apply
Type-Successive ratio.
Ratio-1.05 (Double sided)
Spacing-Apply
Interval Count- 90
Mesh-Apply
Click Apply.

(b) Select edge- QR, SP
Grading-Apply.
Type-Successive Ratio.
Ratio-1.05 (Double sided)
Spacing-Apply
Interval Count-28
Mesh-Apply
Click Apply.

STEP 6 Subtract faces.

Operation>Geometry>Face Command Button>Boolean operations>Subtract.

Subtract face **PQRS** from **ABCD**.

STEP 7 Mesh face.

Operation>Mesh Command Button>Face mesh>Mesh faces.

Faces- Select face 1
Elements- Quad.
Type- Pave
Interval Size- 2

STEP 8 Face Creation and meshing.

Again, create a face **PQRS** by joining the sides **PQ, QR, RS, SR**.
Mesh **PQRS** by following the above meshing method with an interval size of 2. The scheme should be Pave mesh.

STEP 9 Meshing Edge **DO**:

Operation>Mesh>Mesh Edges.

Select Edge-BH
Grading-Apply

Type-Successive ratio.

Ratio-1

Spacing-Apply

Interval Count-20

Mesh-Apply

Click Apply

STEP 10 Creation of Volume:

Operation>Geometry>Volume>Form Volume>Sweep Real Faces.

Select all the faces.

Path-Edge.

Edge-Select Line DO with mesh.

Click Apply.

STEP 11 Creating Zones.

Operation>Zones Command Button>Specify Boundary Types.

<u>Name</u>	<u>Type</u>
Inlet sink	Wall
Channel inlet	Mass flow inlet
Heat flux	Wall
Base channel	Wall
Channel right	Wall
Channel left	Wall
Sink right	Wall
Channel top	Wall
Sink left	Wall
Sink top	Wall
Sink outlet	Wall
Channel outlet	Outflow

STEP 12 Creation of Continuum.

Operation>Zones Command Button>Specify Continuum Types.

<u>Name</u>	<u>Type</u>
Heat sink	Solid
Channel	Liquid

STEP 13 File>Save as [name]

STEP 14 File>export>Mesh[File name]

CHAPTER 4

FLUENT ANALYSIS OF MICROCHANNEL

ANALYSIS OF MICROCHANNEL IN FLUENT:

STEP 1 Select **FLUENT 3ddp**.

STEP 2 Reading of the mesh file.

File>Read>Case.

STEP 3 **ANALYSIS OF GRID.**

a) Checking of grid

Grid>Check.

It was checked that the total volume doesn't come as negative.

b) Scaling of Grid.

Grid>Scale.

Scale was set to 1e-6 in X, Y, Z directions.

c) Smoothing and Swapping.

Grid>Smooth/swap.

The grid was swapped until Zero faces were moved.

STEP 4 **SELECTION OF MODELS.**

(a) Defining solver.

DEFINE>MODELS>SOLVER.

Solver is segregated

Implicit formulation

Space steady

Time steady

(b) Define Energy Equation.

DEFINE > MODEL >ENERGY.

(c) Selection of Materials.

DEFINE > MATERIALS.

Fluid was taken to be liquid water which was selected from the fluent database.

Solid was changed to **SILICON** whose properties were:

Density, = 2330 Kg/m³.

Specific Heat Capacity, k= 714 J/Kg/°C.

Thermal Conductivity, C_p= 140

(d) Defining Operating Conditions.

DEFINE > OPERATING CONDITIONS.

Operating pressure= 101.325 KPa

Gravity = -9.81 m/s² in Y-direction

(e) Defining Boundary Conditions.

1. Channel Inlet.

Input the following values at Mass Flow Inlet.

Mass Flow Rate- 1e-5/ 6e-6 Kg/s

Temperature- 293 K

Component of X and Y velocity Direction- 0

Component of Z velocity Direction- 1

2. Heat Flux.

Input the following values at Wall.

Select Heat Flux from the list.

Heat Flux = 50/90/150 e4 W/m².

Thickness = 450e-6 m

Click on **shell conduction**.

3. Channel left.

Input the following values at Wall.

Select Heat Flux from the list.

Heat Flux = 0 W/m².

Thickness = 21.5e-6 m

Click on **shell conduction**

4. Channel right.

Input the following values at Wall.

Select Heat Flux from the list.

Heat Flux = 0 W/m².

Thickness = 21.5e-6 m

Click on **shell conduction**

5. Channel top.

Input the following values at Wall.

Select Heat Flux from the list.

Heat Flux = 0 W/m².

Thickness = 450e-6 m

Click on **shell conduction**

STEP 5 **SELECTION_OF SOLUTION.**

(a) Defining Solutions.

SOLVE > CONTROLS>SOLUTIONS

Flow and energy equations used.

Pressure-Velocity Coupling- SIMPLE.

Under relaxation factors:

Pressure= 0.3

Density= 1

Body Force= 1

Momentum= 0.08

Energy= 1

Discretization:

Pressure= Standard.

Momentum= 2nd order upwind.

Energy= 2nd order upwind.

(b) Initializing the Solution.

The Problem was computed taking the values from all zones.

- (c) Defining Monitors.

SOLVE > MONITORS>RESIDUALS.

Plot option was clicked.

Plotting was done in window 1 with 1000 iterations.

Convergence criterion:

Continuity= 0.0001

X-velocity= 0.0001

Y-velocity= 0.0001

Z- Velocity= 0.0001

Energy= 1e-8.

- (d) Iterating the solution.

SOLVE >ITERATE.

No. of iterations=500.

Iterate till the solution is converged.

STEP 6 Plotting of Contours and Graphs.

- (a) Temperature Contour:

Display>Contours.

Contours of Temperature (Static Temperature) was selected.

Filled option was clicked.

Channel Outlet surface was selected.

Click Display to see the Contours at Channel Outlet.

- (b) Velocity Contour:

Display>Contours.

Contours of Velocity (Velocity Magnitude) was selected.

Filled option was clicked.

Channel Outlet surface was selected.

Click Display to see the Contours at Channel Outlet

- (c) Nu vs Z Graphs:

Plot>XY

Plot direction, X=0; Y=0; Z=1.

Y-Axis Function- Wall Fluxes> Surface Heat Transfer Co-efficient.
Click Channel Outlet on surfaces and write to MS-EXCEL file.
Calculate Nusselt Number and Plot against Z-direction.

(d) Re vs Z graph:

Defining Custom Field Functions- Reynold's Number.

Define> Custom Field Functions.

Create a function $Re = |V| * D_h$

Where V is the velocity magnitude

D_h = Hydraulic Diameter=0.000086 m

Plot>XY

Plot direction, X=0; Y=0; Z=1.

Y-Axis Function- Custom field function>**Re**.

Click Channel Outlet on surfaces and write to MS-EXCEL file.

Plot Re vs Z in MS-EXCEL file.

(e) Nu vs Re graph:

Create a new MS-EXCEL file.

Copy the Reynolds number at 50/30 kpa.

Copy the Nusselt Number at 50/90/150 W/cm².

Plot graph with Reynolds number in X-axis and Nusselt number in Y-axis.

CHAPTER 5

RESULTS AND DISCUSSIONS

RESULTS

SINGLE CHANNEL:

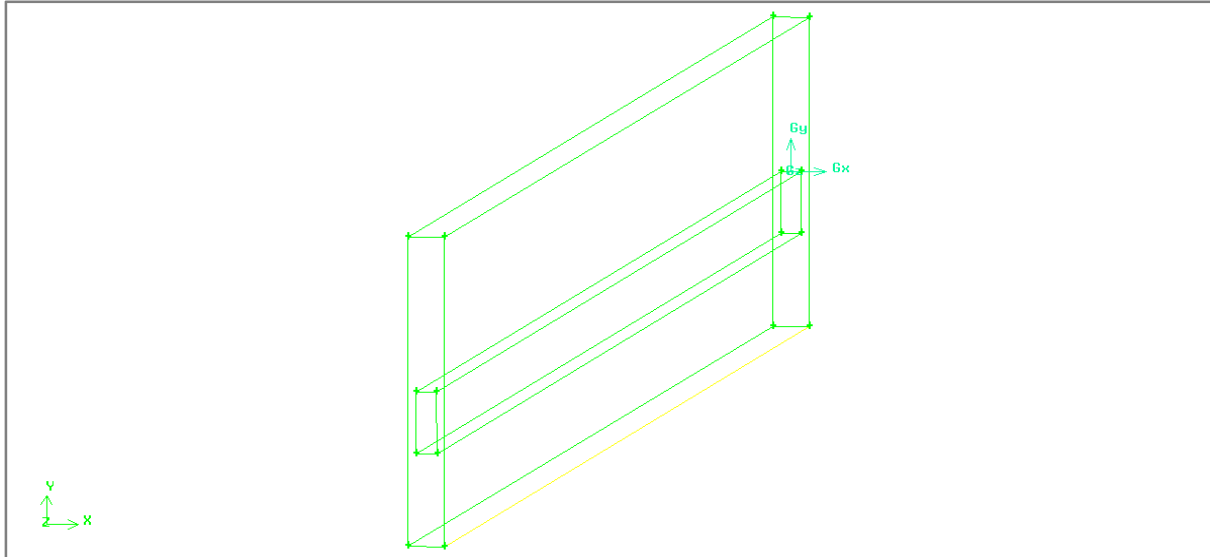


Fig. 2 Schematic diagram of Single Microchannel in GAMBIT.

Outlet conditions at 50 KPa and adiabatic walls:

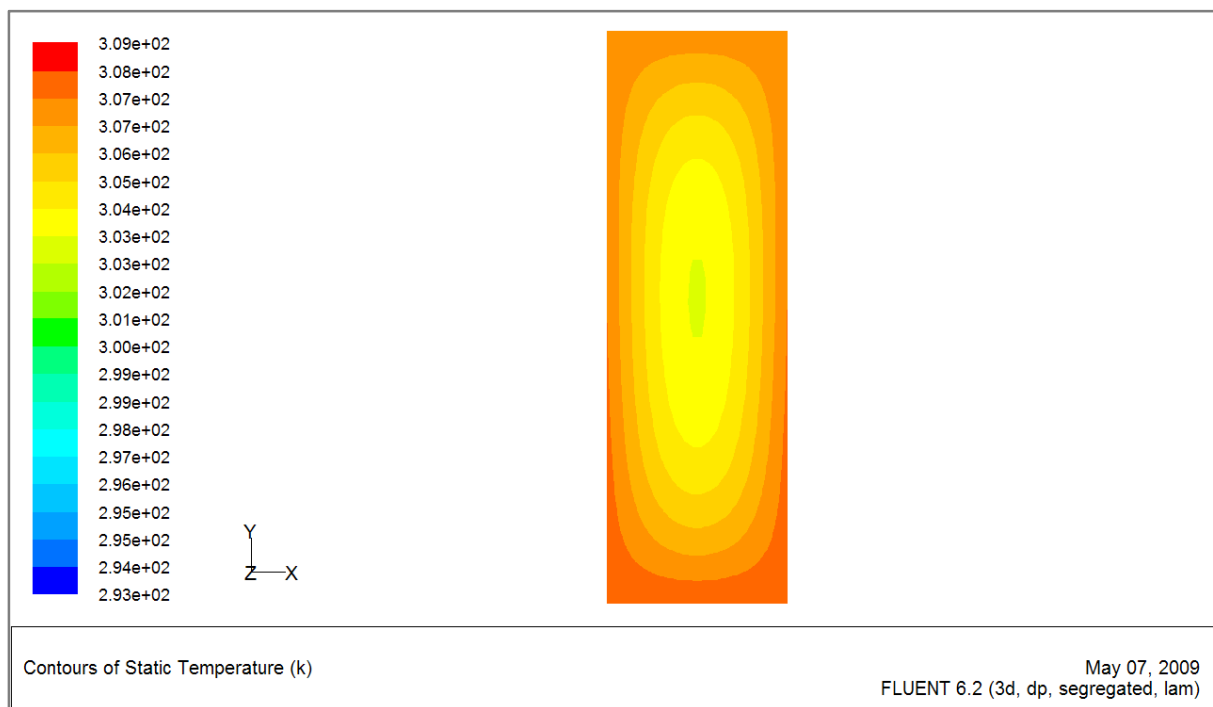


Fig. 3 Temperature contour of Outlet at 50 w/cm²

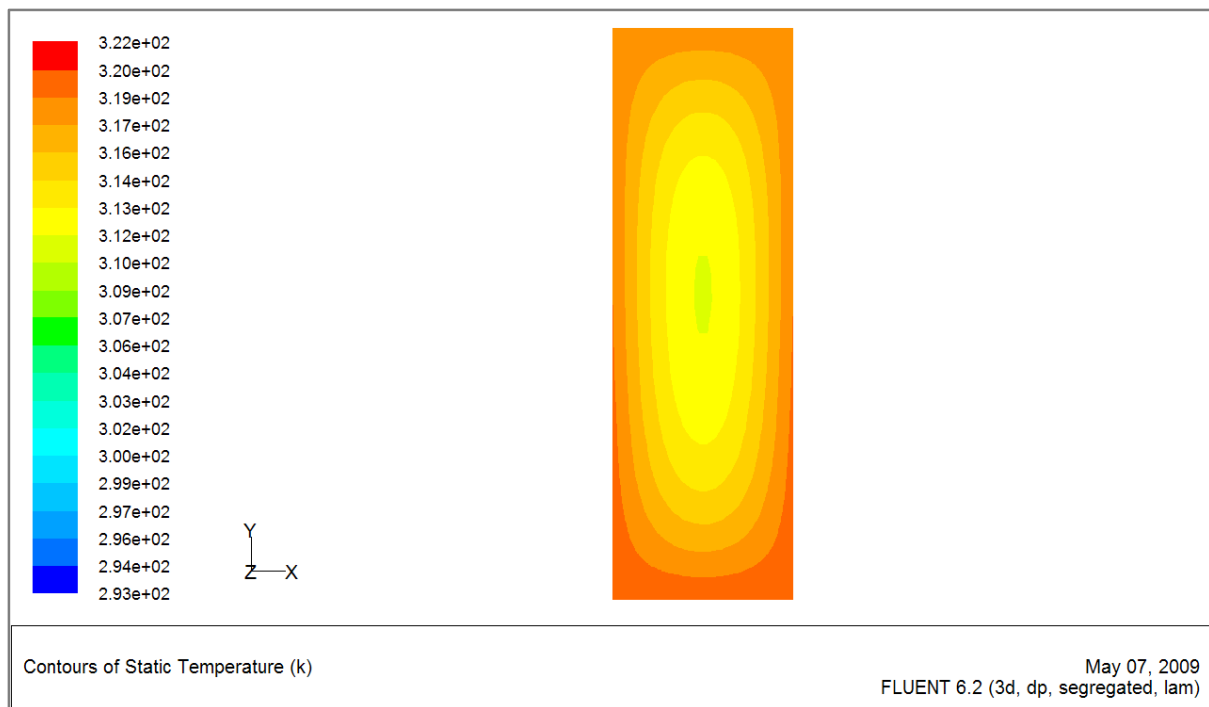


Fig. 4 Temperature of Outlet at 90 w/cm²

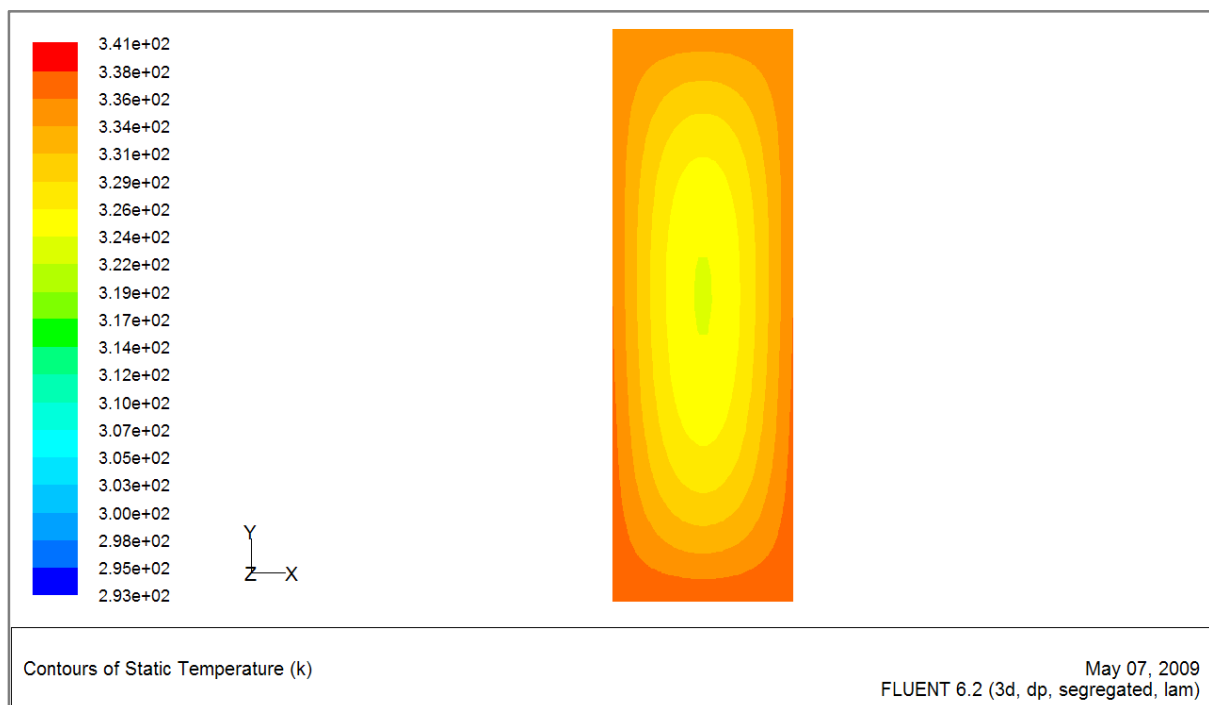


Fig. 5 Temperature of Outlet at 150 w/cm²

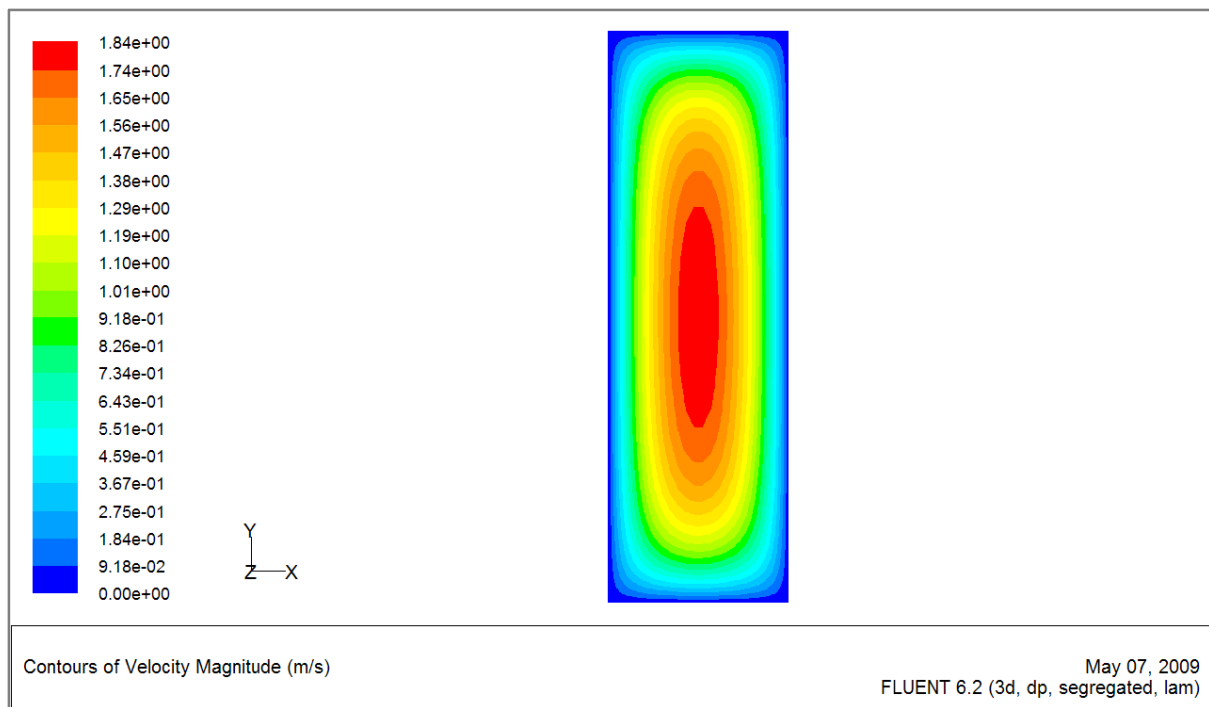


Fig. 6 Velocity contour of outlet at 50 kpa

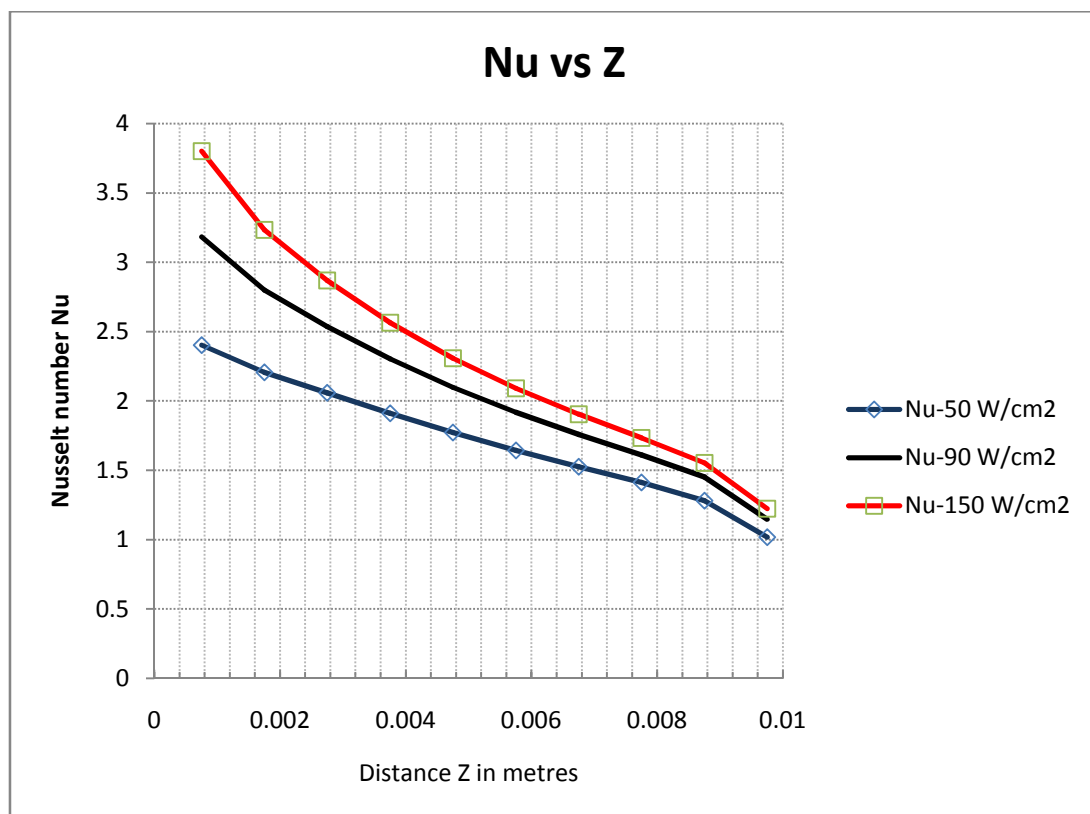


Fig. 7 Combined Nu vs Z

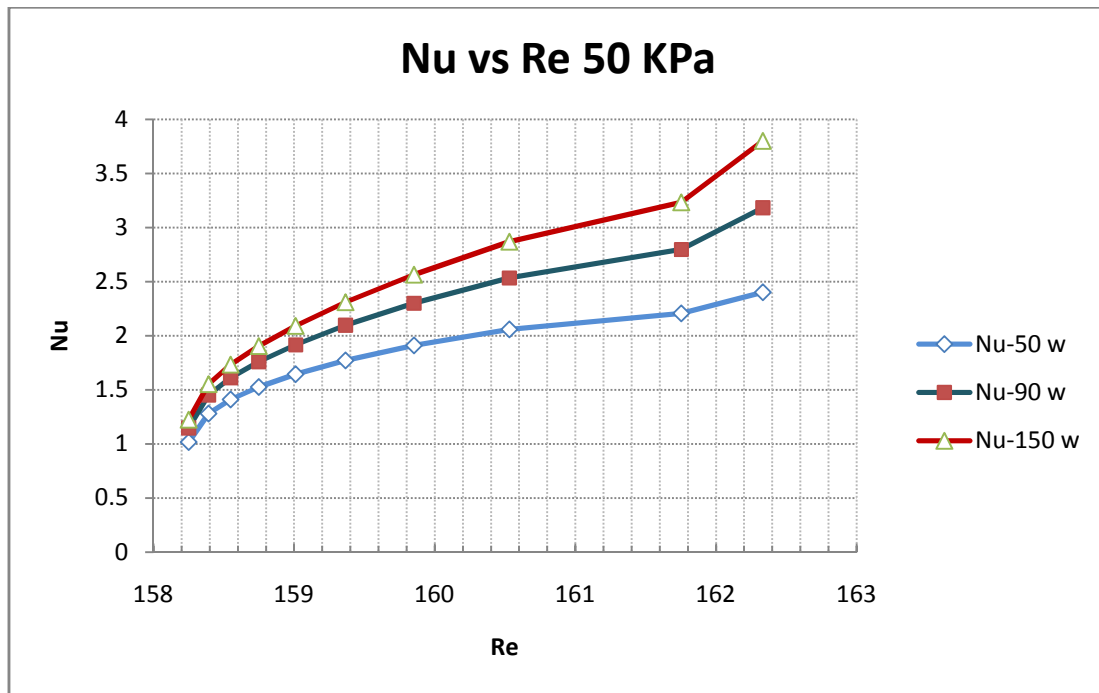


Fig. 8 Nu vs Re at 50 kpa

Outlet conditions at 30 KPa and adiabatic walls:

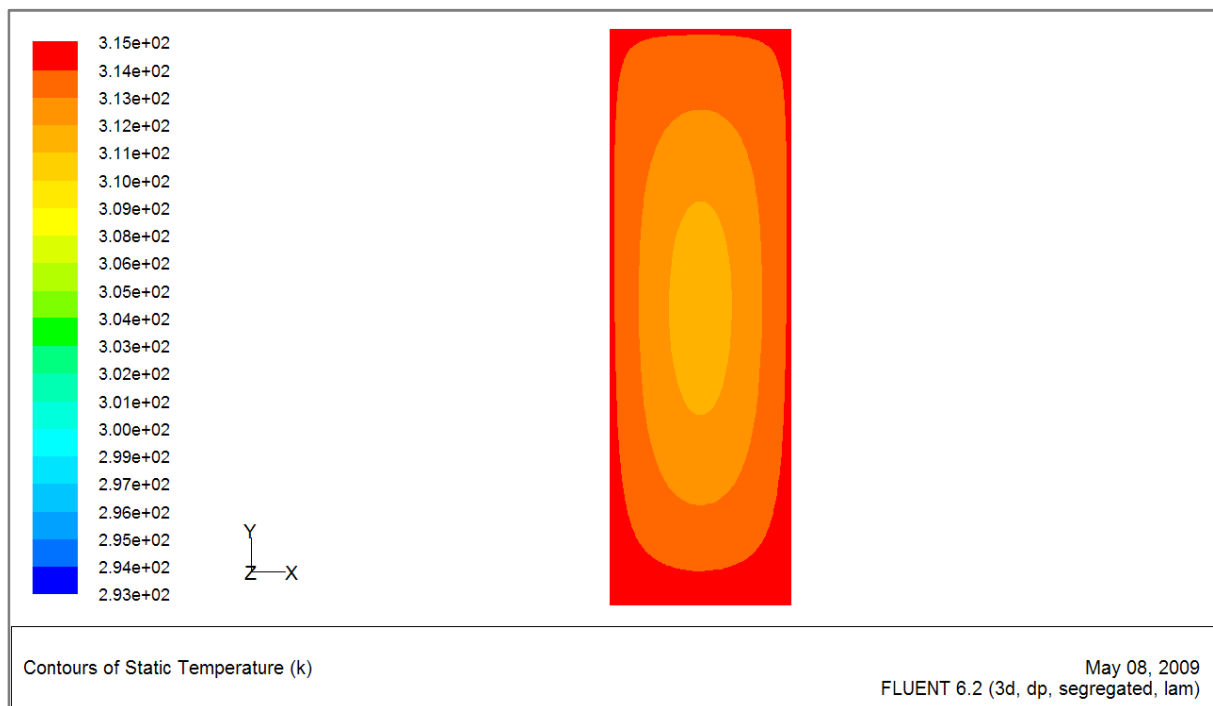


Fig. 9 Temperature of Outlet at 50 w/cm².

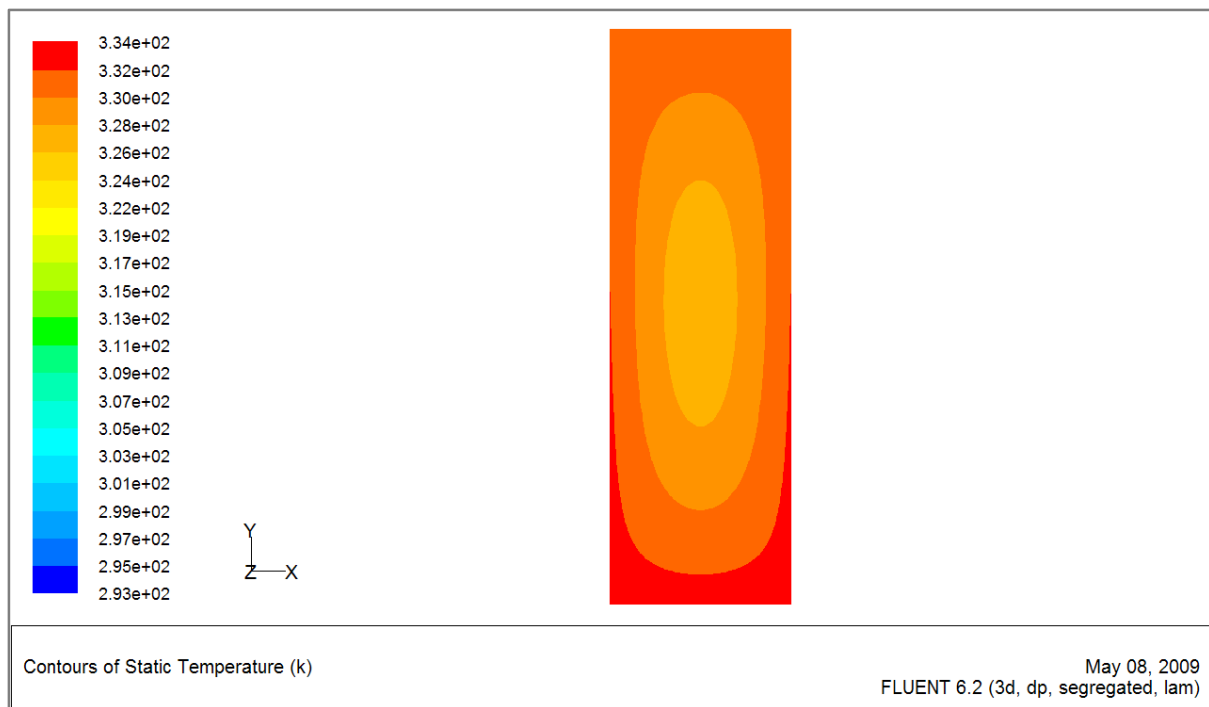


Fig. 10 Temperature of Outlet at 90 w/cm²

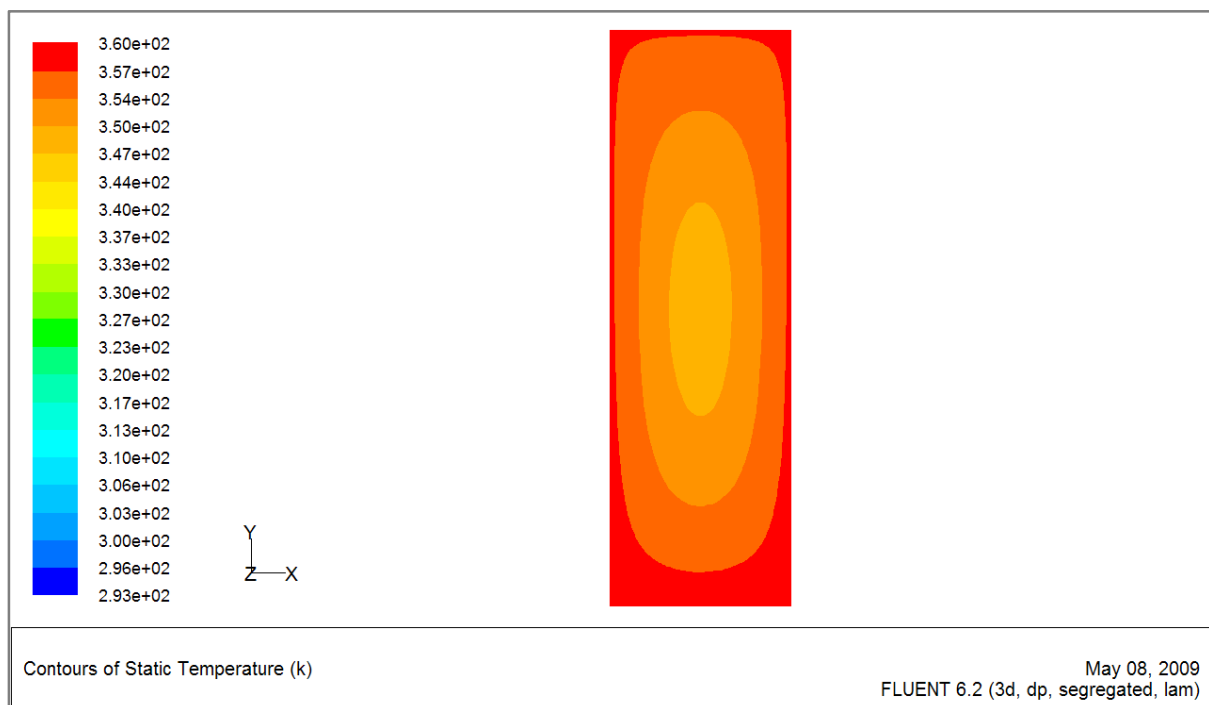


Fig. 11 Temperature of Outlet at 150 w/cm²

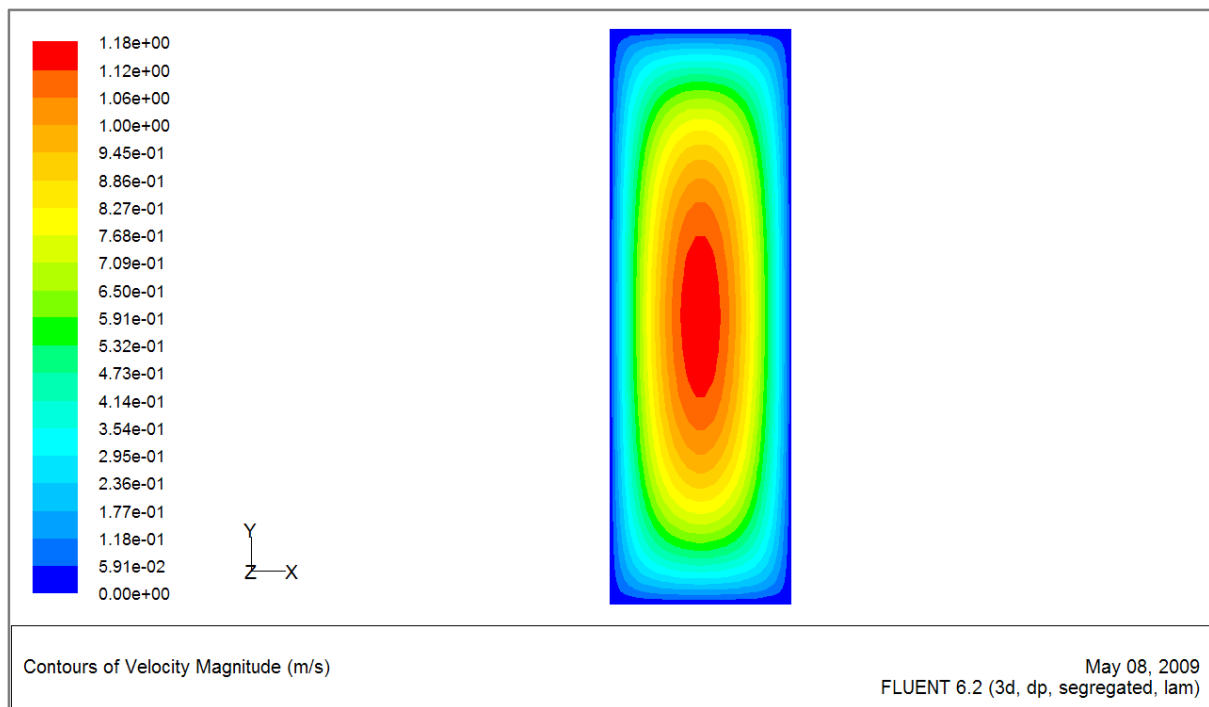


Fig. 12 Velocity contour of outlet at 30 kpa

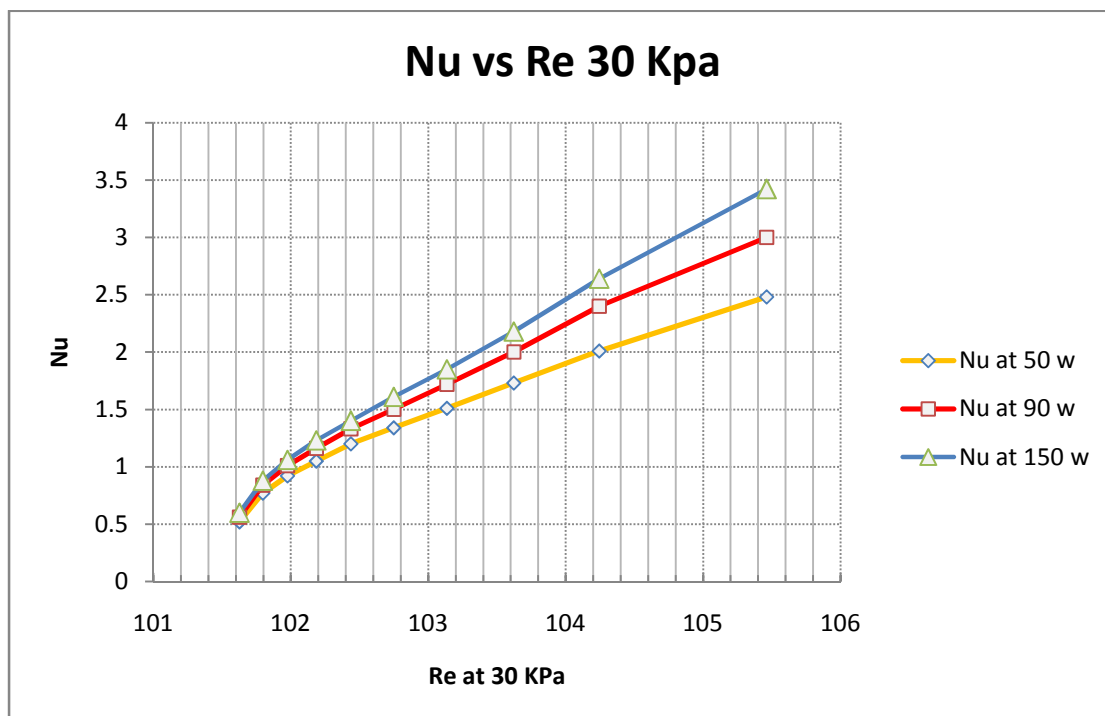


Fig 13 Nu vs Re at 30 KPa

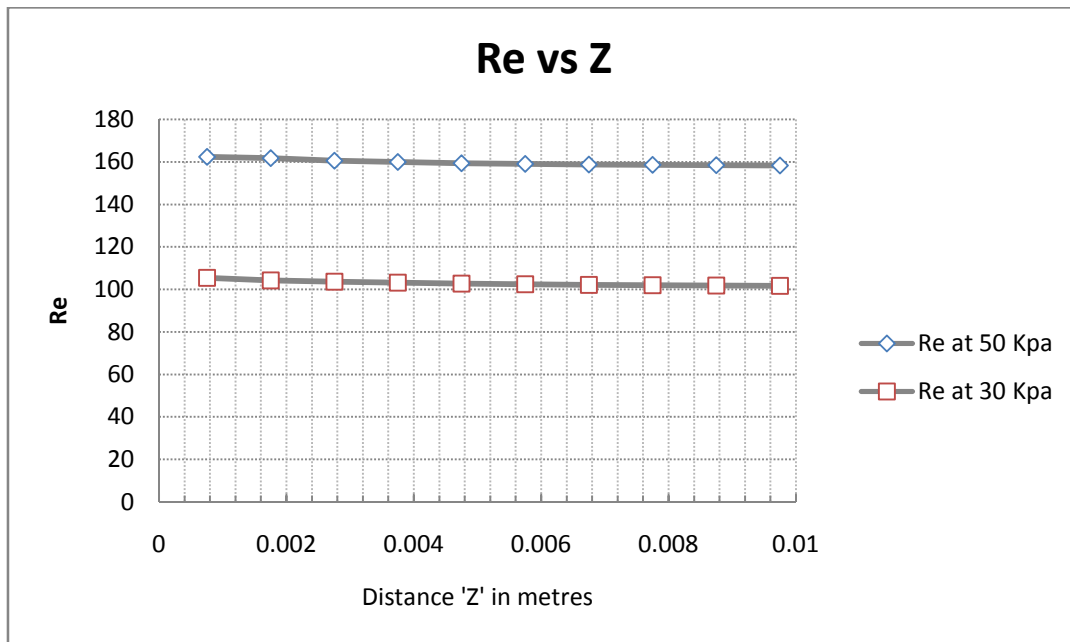


Fig. 14 Combined Re vs Z

DOUBLE CHANNEL:

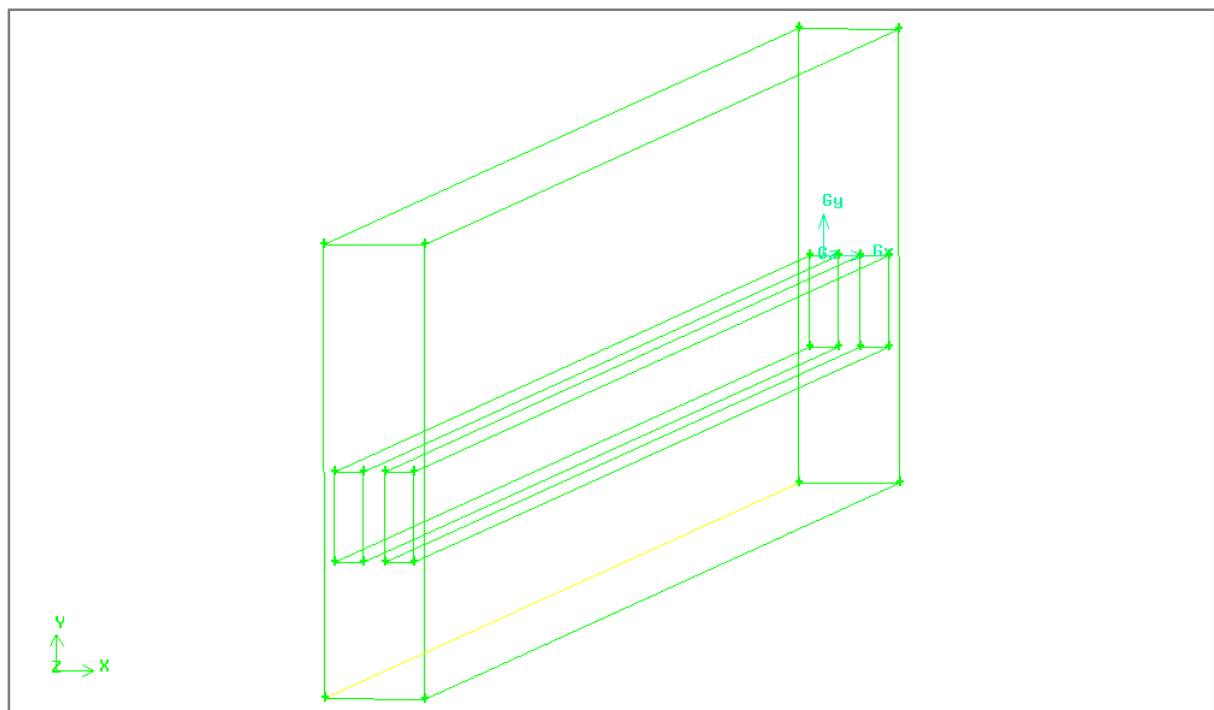


Fig. 15 Schematic Diagram of a Double Micro-channel in GAMBIT.

WATER AT 50 KPa & 50 W/cm²:

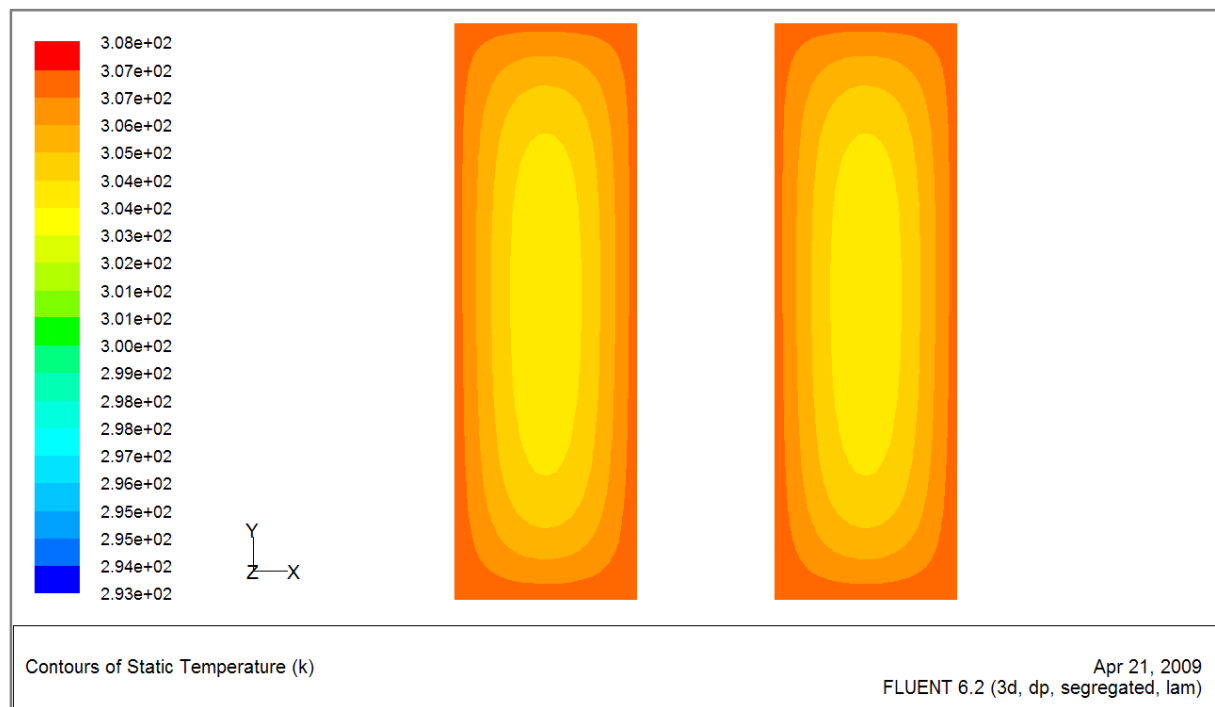


Fig. 16 Temperature of Outlet at Adiabatic wall conditions

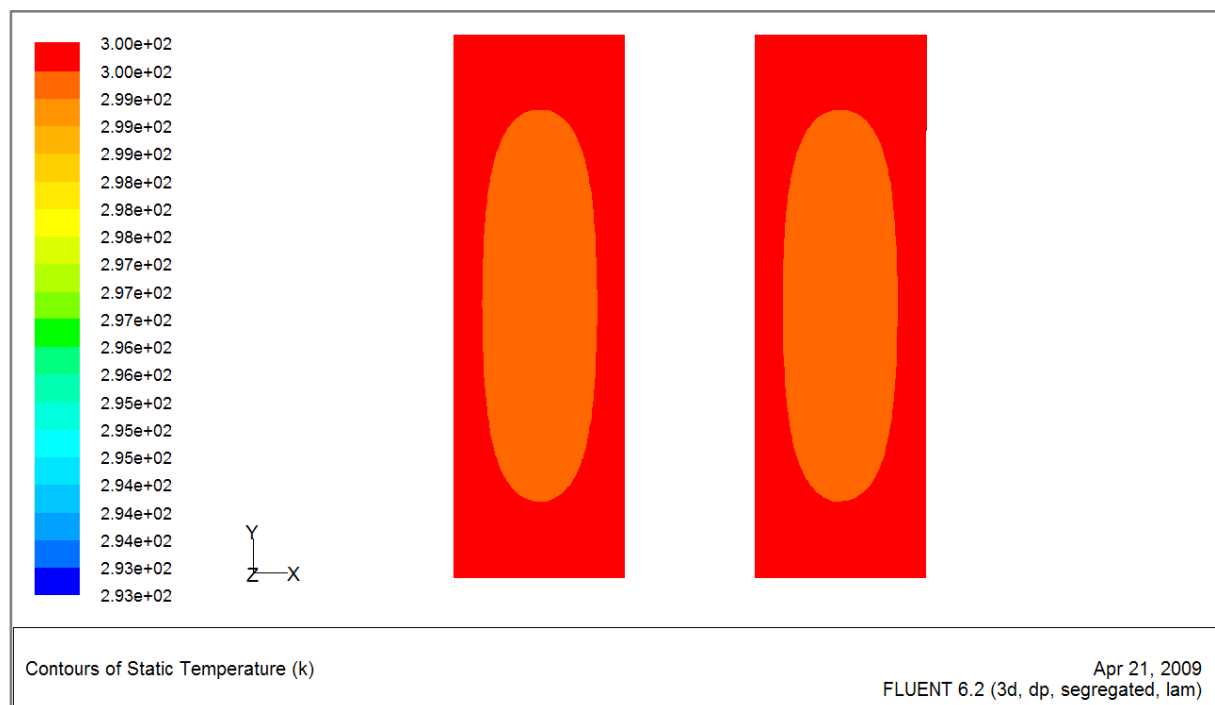


Fig. 17 Temperature contour at Isothermal (300k) wall condition.

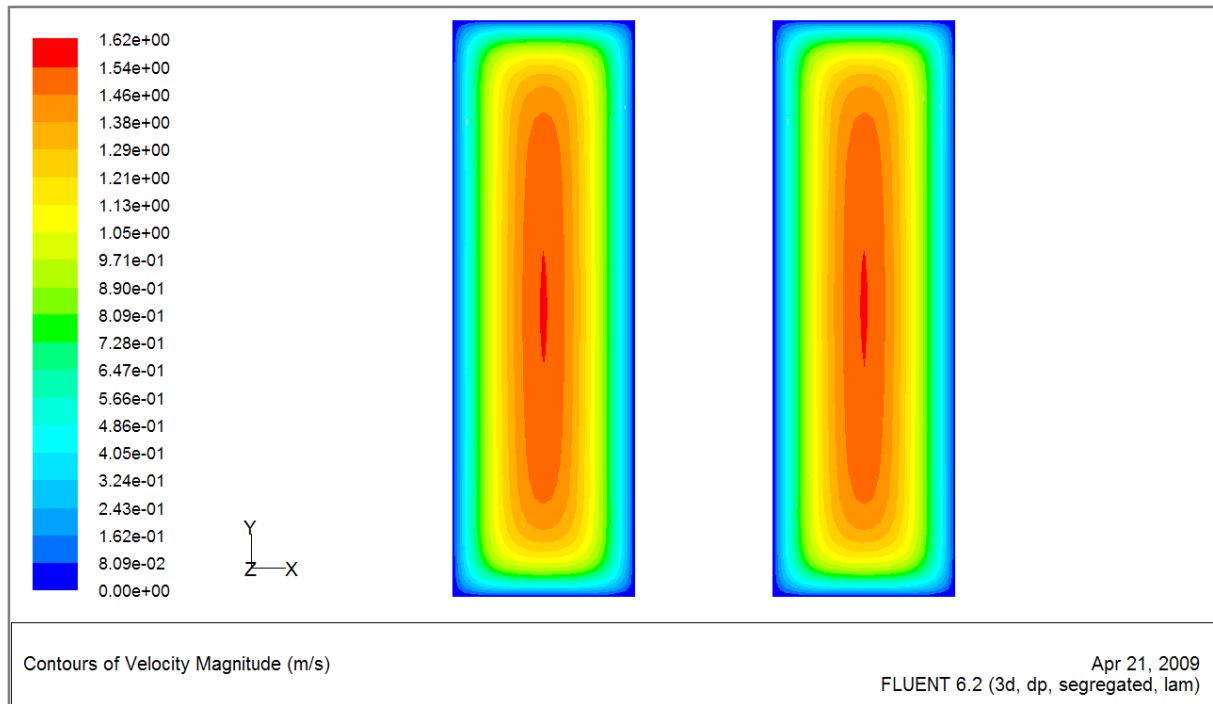


Fig. 18 Velocity contour at outlet (Isothermal & Adiabatic walls)

DISCUSSIONS:

GRID:

Fig 2 and Fig 15 shows the schematic diagram of Single and Double Micro-channel respectively in GAMBIT. The modeling was done in according to the dimensions given. The grid shows the mass flow inlet, outlet as well as the conducting walls of the heat sink and the channels.

CONTOURS OF TEMPERATURE (SINGLE CHANNEL):

Fig 3, Fig 4 and Fig. 5 displays the filled contours of temperature at outlet of the micro-channel for heat fluxes of 50, 90 and 150 w/cm² at a pressure difference of 50 KPa. As the heat flux increases, the outlet temperature goes on increasing because the fluid gets heated up more and more due to convective heat transfer which is evident from the contour profiles.

Fig 9, Fig 10 and Fig 11 displays the filled contours of temperature at outlet of the micro-channel for heat fluxes of 50, 90 and 150 w/cm² at a pressure difference of 30 KPa. As the heat flux increases, the outlet temperature goes on increasing because the fluid gets heated up more and more due to convective heat transfer which is evident from the contour profiles. Also, as the mass flow rate decreases, the temperature of the outlet at a particular heat flux for 50 KPa is higher than that of 30 KPa because less amount of convective heat transfer takes place inside the micro-channel.

$$T_{150} > T_{90} > T_{50} \quad (\text{At 50 Kpa, 30 kPa})$$

$$T_{50, (50 \text{ kPa})} > T_{50, (30 \text{ kPa})}$$

$$T_{90, (50 \text{ kPa})} > T_{90, (30 \text{ kPa})}$$

$$T_{150, (50 \text{ kPa})} > T_{150, (30 \text{ kPa})}$$

CONTOURS OF VELOCITY (SINGLE MICRO-CHANNEL):

Fig 6 and fig 12 displays the velocity of the fluid (water) at the outlet for 50 Kpa and 30 KPa respectively. As the pressure difference decreases, the velocity decreases which is evident from the profiles at outlet with the maximum velocity decreasing from 1.84 m/sec to 1.18 m/sec.

$$V_{50 \text{ KPa}} > V_{30 \text{ KPa}}$$

CONTOURS OF TEMPERATURE (DOUBLE MICROCHANNEL):

In case of a double Micro-channel, at 50 KPa and a heat flux of 50 w/cm² (Fig. 16) the outlet temperature at adiabatic wall conditions (308 k) was much greater than those at Isothermal boundary conditions (300 k) (Fig. 17). It was also found that the outlet temperature of double micro-channel (308 k) (Fig. 16) was lower than the single micro-channel (309 k) (Fig. 3).

$$T_{\text{adiabatic, 50 KPa}} > T_{\text{isothermal, 50 KPa}} \quad (\text{Double Micro-channel})$$

$$T_{\text{adiabatic, double, 50 KPa}} < T_{\text{adiabatic, single, 50 KPa}}$$

CONTOURS OF VELOCITY (DOUBLE MICRO-CHANNEL):

The velocity of the fluid was found to be 1.62 m/sec at the outlet for a pressure difference of 50 Kpa as shown by Fig. 18.

GRAPHS:

1. Nu vs Z Graph:

From the graph of Fig. 7, the Nusselt number at a particular heat flux keeps on decreasing with respect to distance covered in the micro-channel. This is mainly due to the decrease in convective heat transfer rate as the fluid gets heated up more and more and is unable to carry that much amount of heat.

Also, it can be noticed (Fig. 7) that the nusselt number at a particular point in the micro-channel is different for different heat fluxes and decreases with decreasing heat flux

$$Nu_{150} > Nu_{90} > Nu_{50} \quad (\text{At } 50 \text{ KPa} \text{ \& } 30 \text{ KPa})$$

2. Re vs Z Graph:

Reynold's number from the graph (Fig. 14) was found to lie in the laminar region which proved that the value calculated was correct. It was also found that as the pressure difference decreases from 50 KPa to 30 KPa, the Reynold's number also decreased with respect to distance.

$$Re_{50 \text{ KPa}} > Re_{30 \text{ KPa}}$$

3. Nu vs Re Graph:

As Reynold's number increased, Nusselt number was also found to increase, both in case of 50 KPa (Fig.8) and 30 KPa (Fig. 13). The slope of the graph gradually decreased as heat flux decreased from 150 W/cm² to 50 W/cm². This was found to be true in both cases of 50 KPa and also 30 KPa.

6. CONCLUSION:

The analysis performed, provides a fundamental understanding of the combined flow and conjugate convection–conduction heat transfer in the three-dimensional micro-channel heat sink. The model formulation is general and only a few simplifying assumptions are made. Therefore, the results of the analysis as well as the conclusions can be considered as quite general and applicable to any three-dimensional conjugate heat transfer problems.

- A three-dimensional mathematical model, developed using incompressible laminar Navier–Stokes equations of motion, is capable of predicting correctly the flow and conjugate heat transfer in the micro-channel heat sink. It has been validated using experimental data reported in the literature, and a good agreement has been found between the model predictions and measurements.
- The combined convection–conduction heat transfer in the micro-channel produces very complex three-dimensional heat flow pattern with large, longitudinal, upstream directed heat recirculation zones in the highly conducting silicon substrate as well as the local, transverse heat ‘vortices’ in the internal corners of the channel where the fluid and solid are in direct contact. In the ‘vortex’ region, the local surface heat fluxes and the local convective heat transfer coefficients (Nusselt number) become negative because the bulk (mixed mean) temperature is not an appropriate reference temperature for describing the heat flow direction locally everywhere. In other words, the concept of the local heat transfer coefficient and Nusselt number is meaningless in the strongly conjugate problems.
- Although only the right Z-wall outside the channel is heated, the heat is redistributed by conduction within a substrate and is transferred to the coolant through all four walls inside the channel.
- The local heat fluxes from the solid to the coolant in the small inlet region of the micro-channel are larger than those in the further downstream portion by more than two orders of magnitude. This is because the average convective heat transfer coefficient is much larger in the upstream locations (the boundary layer thickness is small) and also

because the highly conducting channel walls support very effective heat redistribution from the downstream (large convective resistance) to the upstream (small convective resistance) regions of the channel. This finding supports the concept of the manifold micro-channel (MMC) heat sink where the flow length is greatly reduced to small fraction of the total length of the heat sink by using a design with multiple inter-connected inlets and outlets.

7. APPLICATION OF MICROCHANNEL HEAT SINKS:

Theoretical analysis and experimental data strongly indicate that the forced convection water cooled micro-channel heat sink has a superior potential for application in thermal management of the electronic packages. The heat sink is compact and is capable of dissipating a significant thermal load (heat fluxes of the order 100 W/cm^2) with a relatively small increase in the package temperature (less than 20°C), if operated at the Reynolds numbers above 150.

BIBLIOGRAPHY

1. Tuckerman, D.B. and Pease, R.F.W., High-performance heat sinking for VLSI, IEEE Electron Device Letters, 1981
2. Wu, P. and Little, W.A., Measurement of friction factors for the flow of gases in very fine channels used for microminiature Joule-Thomson refrigerators, Cryogenics
3. Peng, X.F., Peterson, G.P., and Wang B.X., Heat transfer characteristics of water flowing through microchannels, Experimental Heat Transfer, 1994
4. Peng, X.F. and Peterson, G.P., Effect of thermofluid and geometrical parameters on convection of liquids through rectangular microchannels, International Journal of Heat and Mass Transfer, 1995, Vol. 38(4),
5. Judy, J., Maynes, D., and Webb, B.W., Characterization of frictional pressure drop for liquid flows through microchannels, International Journal of Heat and Mass Transfer, 2002
6. Liu, D. and Garimella, S.V., Investigation of liquid flow in microchannels, AIAA Journal of Thermophysics and Heat Transfer, 2004, Vol. 18, p. 65–72
7. Choi, S.B., Barron, R.F., and Warrington, R.O., Fluid flow and heat transfer in microtubes, Micromechanical Sensors, Actuators, and Systems, ASME DSC, 1991, Vol. 32, p. 123–134
8. Rahman, m.m.and gui. Experimaental measurements of fluid flow and heat transfer in microscale cooling passage in a chip susstrate.
9. A.G. Fedorov, R. Viskanta, Three-dimensional conjugate heat transfer in the microchannel heat sink for electronic packaging, Int. J. Heat Mass Transfer 43 (3) (2000)
10. FLUENT documentation.
11. Poh-Seng Lee, Suresh V. Garimella, Thermally developing flow and heat transfer in rectangular microchannels of different aspect ratios, International Journal of Heat and Mass Transfer 49 (2006) 3060–3067
12. www.wikipedia.com
13. Numerical Heat Transfer and Fluid Flow by Suhas V. Patankar
14. Computational fluid Dynamics by John D. Anderson.